WAVECAD TUTORIAL

3151

DEMONSTRATION

Publication No. 980788

Racal Instruments

RACAL

WAVECAD TUTORIAL

3151

DEMONSTRATION

Publication No. 980788

RACAL INSTRUMENTS

Racal Instruments, Inc.
4 Goodyear St., Irvine, CA 92618-2002
Tel: (800) RACAL-ATE, (800) 722-2528, (714) 859-8999; FAX: (714) 859-7139

Racal Instruments, Ltd.
480 Bath Road, Slough, Berkshire, SL1 6BE, United Kingdom
Tel: +44 (0) 1628 604455; FAX: +44 (0) 1628 662017

Racal Systems Electronique S.A.
18 Avenue Dutartre, 78150 LeChesnay, France
Tel: +33 (1) 3955 8888; FAX: +33 (1) 3955 6735

Racal Systems Elettronica Srl.
Strada 2-Palazzo C4, 20090 Milanofiori Assago, Milan, Italy
Tel: +39 (2) 5750 1796; FAX +39 (2) 5750 1828

Racal Elektronik System GmbH.
Frankenforster Strasse 21, 51427 Bergisch-Gladbach 1, Germany
Tel:+49 2204 92220; FAX: +49 2204 21491

Racal Australia Pty. Ltd.
3 Powells Road, Brookvale, NSW 2100, Australia
Tel: +61 (2) 936 7000, FAX: +61 (2) 936 7036

Racal Electronics Pte. Ltd.
26 Ayer Rajah Crescent, 04-06/07 Ayer Rajah Industrial Estate, Singapore 0513.
Tel: +65 7792200, FAX: +65 7785400

Racal Inst Racal Instruments, Ltd.
Sun House 13th Floor, 181 Des Voeux Road, Central, Hong Kong
Tel: +852 281 58633, FAX +852 281 58141

RACAL

PUBLICATION DATE: JUNE 1996

Copyright © 1996 by Racal Instruments, Inc. Printed in the United States of America. All rights reserved. This book or parts thereof may not be reproduced in any form without written permission of the publisher.

TABLE OF CONTENTS

Getting Started..... 1
System Requirements..... 1
Installing WaveCAD and the Demonstration Files 2
Using WaveCAD to Control a Model 3151..... 2
Creating Waveforms Using WaveCAD 4
Displaying a Standard Waveform 4
Modifying a Standard Waveform 5
Loading a Waveform From a File 6
Downloading Waveforms to a Model 3151..... 6
Adding Noise to Waveforms 7
Adding Noise to a Digital Waveform..... 7
Adding Noise to an Analog Waveform 8
Loading a Waveform From a File 9
Downloading Noise Waveforms to a 3151..... 10
Creating Shift-Keyed Waveforms..... 11
Creating an ASK Waveform..... 11
Creating an FSK Waveform..... 12
Creating a PSK Waveform..... 13
Loading a Shift-Keyed Waveform From a File..... 13
Downloading to a Model 3151..... 14
Creating Modulated Waveforms..... 15
Creating an AM Waveform..... 15
Creating a PM Waveform..... 16
Loading a Modulated Waveform..... 17
Downloading to a Model 3151..... 17
Creating Swept Waveforms 19
Creating a Linear Sweep Waveform 19
Creating a Logarithmic Sweep Waveform..... 20
Loading a Swept Waveform..... 21
Downloading to a Model 3151..... 21
Creating Multi-Phase Waveforms 23
Displaying a Multi-Phase Waveform 23
Modifying a Multi-Phase Waveform 23
Loading a Waveform From a File 23
Downloading to Multiple Model 3151's..... 24
Creating Sequenced Waveforms..... 26
Creating a Sequence..... 26
Downloading a Sequence 27
Glossary..... 28
Index 30

This section explains the hardware and software system requirements to run this demonstration, how to install the demonstration and how to use WaveCAD to control a Model 3151.

You may use this demonstration Using WaveCAD either as a software only demonstration on a computer, or as a hardware/software demonstration requiring VXIbus hardware. Demonstrations follow which may be used with or without 3151 hardware. Demonstrations using hardware are also included. Software only demonstrations should skip such sections.

System Requirements

The software only demonstration requires the following hardware:

- 80486 33 MHz computer, minimum.
- 4 Megabytes of RAM.
- 1 Megabyte of available hard disk space (to install the demonstration).
- A mouse or other pointing device.
- A VGA display system with 640x480 resolution, a minimum of 800x600 resolution is preferred.
- Microsoft Windows operating system version 3.1 or better.

The hardware demonstration requires all the above items plus the following:

- One (or more) Model 3151 VXIbus Waveform Generator(s)
- One VXIbus mainframe
- One of the following types of Slot 0 controllers:
  a) Embedded controller (which can also meet the above computer requirements)
  b) GPIB Slot 0 controller (controlled by an external computer with a GPIB card installed)
  c) MXIbus Slot 0 controller (controlled by an external computer with a MXIbus interface card installed)
- A two channel oscilloscope for viewing waveforms (3 channels required for the Multi-Phase Waveform section)
- An appropriate BNC cable for connecting the 3151 to an oscilloscope
- VXIbus software must be installed on your computer or embedded controller as follows:
  a) A VXIbus resource manager (provided with slot 0 controller).
  b) VISA.DLL libraries (v1.0 or better) that support the slot 0 controller being used (also provided with newer controllers). If your slot 0 controller did not come with VISA drivers, contact the manufacturer to obtain the latest drivers.

b) VISA.DLL libraries (v1.0 or better) that support the slot 0 controller being used (also provided with newer controllers). If your slot 0 controller did not come with VISA drivers, contact the manufacturer to obtain the latest drivers.

Installing WaveCAD and the Demonstration Files

First, install WaveCAD for Windows or WaveCAD Demo for Windows (revision 2.0C or higher) from the distribution disk provided. To install WaveCAD:

- Place the WaveCAD distribution disk in the A: drive.
- Use the Windows **Run** feature (either in the Program Manager's **File** menu or under the **Start** menu in Windows 95).
- Enter **A:\SETUP** in the **Command Line** field and press enter.
- Follow the instructions specified in the WaveCAD setup program.

Next, install the 3151 Demonstration by doing the following:

- Place the 3151 Demonstration disk in the A: drive.
- Use the Windows **File/Run** feature in the Program Manager.
- Enter **A:\SETUP** in the **Command Line** field and press enter.
- Follow the instructions in the automatic installation process.

Using WaveCAD to Control a Model 3151 (H/W demo only)

If you have a complete hardware setup to demonstrate the 3151, you are ready to configure the system to run WaveCAD such that it controls the 3151. Follow the following procedure to configure:

- The default configuration of WaveCAD and the Model 3151 is VXIbus Logical Address 2. If another module in your system is using address 2, you may select another address up to 254. If address 255 is used, the resource manager will assign the 3151 a logical address. Otherwise, before installing the 3151 into the chassis, take note of the address you are using.
- Install the Model 3151 into the VXIbus chassis (with slot 0 controller) and turn on power.
- Run a VXIbus "Resource Manager" and verify that a module is found at the appropriate logical address.
- If the module is not found, power down and verify that the logical address dip switches on the side of the 3151 are set to the proper logical address.
- Run WaveCAD.
- Select **VXI/Address and Options...** from the WaveCAD menu bar.
- If the 3151 was set to a logical address other than 2, change it by setting the **Address** field to the chosen logical address.

dialog box is set to the same logical address detected by the resource manager.

This section describes first how to use one of WaveCAD's standard waveform functions to create a waveform. Next, a technique is given to modify this waveform using a user-defined equation. Additionally, a procedure is given for loading a waveform from a previously created waveform file. And when demonstrating WaveCAD with a complete 3151 hardware setup, a procedure is presented which may be used to download a waveform to the waveform generator and view it with an oscilloscope.

The WaveCAD toolbar, which is located below the WaveCAD graph, contains icons which are referred to in this and other demonstration text. If you are unfamiliar with these icons, locate your mouse cursor above each icon. The WaveCAD status bar will list a description of the icon you are examining. The meaning of each icon is shown below:

- Load Waveform File
- Save Waveform
- Save Waveform As
- Draw Freehand Waveform
- Straight Line Segments
- Smooth Marked Area with Spline Function
- Moving Average Filter
- Invert Waveform Between Anchors
- Remove Anchors
- Undo Last Draw
- Import Waveform Between Anchors
- Export Waveform Between Anchors
- Delete All Segments
- Waveform/Parameter Download
- Download Current Sequence

Displaying a Standard Waveform

For this example, WaveCAD is used to create a 1024 point square wave with a 10 Volt peak to peak amplitude. The procedure for doing this is as follows:

- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 1024.
- Click on the **OK** button.
- Select **Wave/Square** from the WaveCAD menu bar.

- Change the **Amplitude** parameter to 10.0.
- Click on the **OK** button.

Modifying a Standard Waveform

Suppose, now, that you would like to modify your standard waveform to mimic the output of a non-ideal square wave source. An example of this uses WaveCAD's Equation feature to create this non-standard waveform as follows:

- In the upper left and right corners of the WaveCAD graph there are triangular shaped objects (called Anchors) pointing inward.
- Drag the right anchor to the position: X = 511 (readback in upper right corner).
- Select **Wave/Equation** from the WaveCAD menu bar.
- Enter the equation **Ampl(p)= sin(10*omg*t) / (0.3*t+1)**.
- In the **Level Adjuster**, set **Amplitude** to 3 and **Offset** to 5.4.
- Click on the **Adjust** button.
- Click on the **Generate** button.
- Click on the **Remove Anchor** icon (to return the right anchor to its home position) and drag the left anchor to position: X = 512.
- Select **Wave/Equation** from **WaveCAD's pull down menu.**
- Enter the equation: **Ampl(p)= sin(10*omg*t+pi) / (0.3*t+1)**.
- In the **Level Adjuster**, set **Amplitude** to 3 and **Offset** to -5.4.
- Click on the **Adjust** button.
- Click on the **Generate** button.
- Click on the **Remove Anchors** icon.

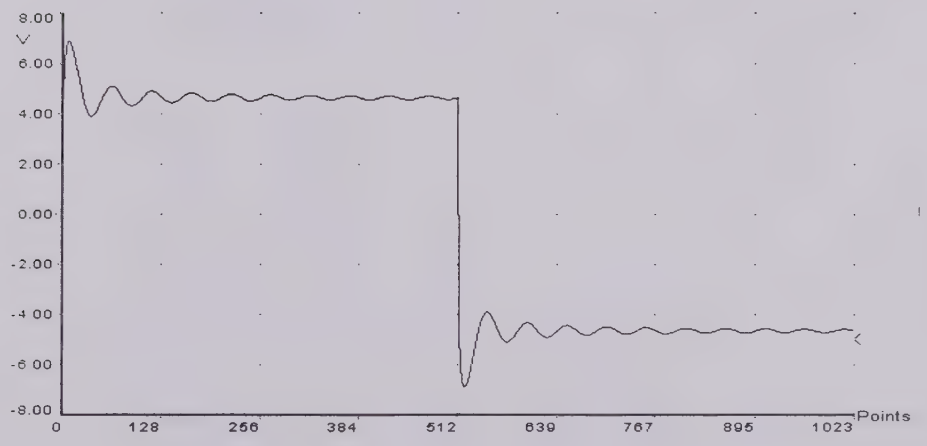

Figure 1: Example Modified Standard Waveform

Loading a Waveform From a File

To load a previously saved version of the modified waveform which was created above:

- Select **File/Open...** from the WaveCAD menu bar.
- Select **RINGPULS.CAD** from the list box and click on **OK**.
- A waveform similar to what was created above is displayed.

Downloading Waveforms to a Model 3151 (H/W Demo only)

To download the waveform displayed on the WaveCAD graph:

- Click on the **Waveform/Parameter Download** icon.
- Select **Setup/Instrument Configuration** from the WaveCAD menu bar.
- Set the Mode to **User Mode**.
- Check the box next to **Output on/off** to turn on the 3151's output.
- Click on the **Download** button.
- Click on the **Close** button.

A repetitive waveform identical to that displayed in the WaveCAD graph will be displayed on the oscilloscope. Be sure to adjust the frequency and amplitude controls on the scope to view the waveform properly.

This section describes how to add noise to a standard waveform. One digital and one analog signal are presented as examples. Additionally, a procedure is given for loading noisy waveforms from previously created waveform files. And, for demonstrating WaveCAD with a complete 3151 hardware setup, a procedure is presented which may be used to download these waveforms to the waveform generator for viewing with an oscilloscope.

Adding Noise to a Digital Waveform

We start using standard waveforms to create a digital signal as follows:

- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Level Min.** to -2, **Level Max.** to 6 and **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Square** from the WaveCAD menu bar.
- Change the **Cycles** to 10, **Amplitude** to 5 and **Offset** to 2.5.
- Click on the **OK** button.
- Move the left anchor to 838 and the right anchor to 1771.
- Click on the **Draw Straight Line Segment** icon.
- Move the left anchor to 5257 and the right anchor to 6247.
- Click on the **Draw Straight Line Segment** icon.
- Move the left anchor to 8723 and the right anchor to 9789.
- Click on the **Draw Straight Line Segment** icon.
- Click on the **Remove Anchors** icon.
- Select **File/Save As...** from the WaveCAD menu bar.
- Set the **Filename** field to **DIGSIG.CAD** click on the **OK** button.

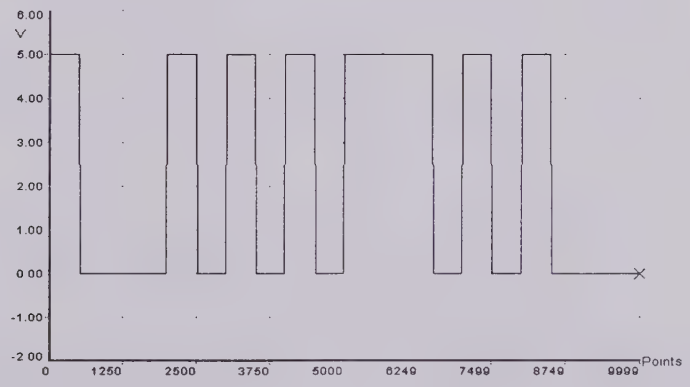

Figure 2: Sample Digital Signal in WaveCAD

For this example, high frequency noise will be added to this digital

waveform. To simulate high frequency noise, WaveCAD's built-in noise function is used. If custom noise waveforms are desired (such as bandlimited noise), please refer to the "VXIbus Applications, Technical Notes and Articles" booklet available from Racal Instruments for instructions on how to create custom noise waveforms for use in WaveCAD.

To use the standard noise waveform to create high frequency noise, do the following:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Noise...** from the WaveCAD menu bar.
- Change the **Amplitude** to 0.3 Volts and click on **OK**.
- Select **File/Save As...** from the WaveCAD menu bar.
- Set the **Filename** field to **HFNOISE.CAD** click on the **OK** button.

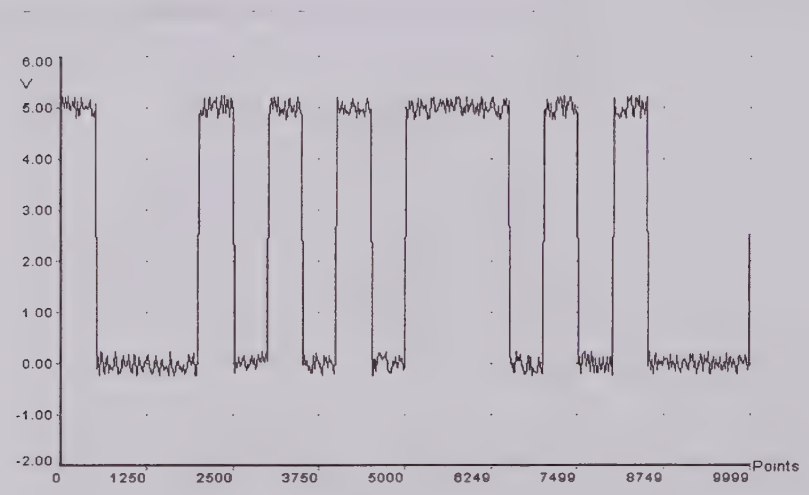

Figure 3: Digital Signal with High Frequency Noise Added

Adding Noise to an Analog Waveform

We are now ready to add the high frequency noise to our digital signal:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Points** to 10000.
- Click on the **OK** button.
- Select **Wave/Equation...** from the WaveCAD menu bar.

- Enter the equation **Ampl(p)= DIGSIG.WAV+HFNOISE.WAV**.
- Click on the **Generate** button.
- If you would like to save your work, use **File/Save As...** from the WaveCAD menu bar.

We are now ready to add the high frequency noise to an analog signal:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 10 and **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Equation...** from the WaveCAD menu bar.
- Enter the equation:
  Ampl(p) = 1500*sin(3*omg*t) + HFNOISE.WAV.
- Click on the **Generate** button.

Note: The sine function was multiplied by a factor of 1500 to scale it for use in units of points. The sine function alone goes from -1 to 1 while the scaling factor converts it to -1500 to 1500. The **HFNOISE.WAV** term is a file that is already stored in units of points. This is because WaveCAD uses point format to store all waveforms; even though the WaveCAD display and Model 3151 both display and output in units of Volts.

Next, re-enter the equation differently to increase the noise:

- Select **Wave/Equation...** from the WaveCAD menu bar.
- Enter the equation:
  Ampl(p) = 1500*sin(3*omg*t) + 2*HFNOISE.WAV.
- Click on the **Generate** button.
- If you choose to save your work, use **File/Save As...** from the WaveCAD menu bar.

Loading a Waveform From a File

To load a previously saved version of a noisy digital waveform like what was created above:

- Select **File/Open...** from the WaveCAD menu bar.
- Select **HFDNOISE.CAD** from the list box and click on **OK**.
- A digital signal with high frequency noise is displayed.

To load a previously saved version of a noisy analog waveform like what was created above:

- Select **File/Open...** from the WaveCAD menu bar.

- Select **HFANOISE.CAD** from the list box and click on **OK**.
- An analog signal with high frequency noise is displayed.

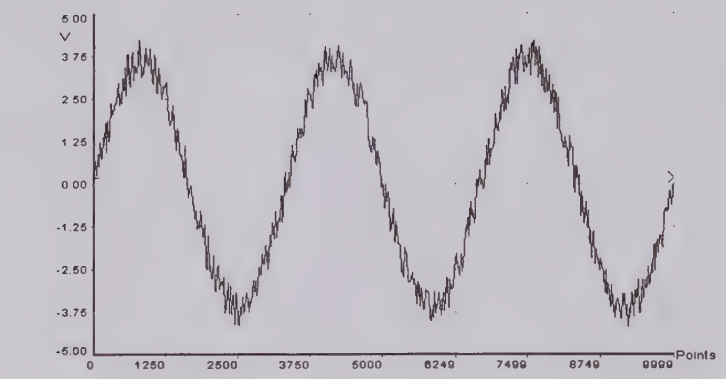

Figure 4: Sine Wave with High Frequency Noise

Downloading Noise Waveforms to a 3151 (H/W Demo only)

To download the waveform displayed on the WaveCAD graph:

- Click on the **Waveform/Parameter Download** icon.
- Select **Setup/Instrument Configuration** from the WaveCAD menu bar.
- Set the **Output/Mode** to **User**.
- Set the **Segment** to 1.
- Check the box next to **Output on/off** to turn on the 3151's output.
- Click on the **Download** button.
- Click on the **Cancel** button.

A repetitive waveform identical to that displayed in the WaveCAD graph will be displayed on the oscilloscope. Be sure to adjust the frequency and amplitude scope controls to view the waveform properly.

Shift-keying is a form of modulation which uses a digital signal to control either amplitude or angular modulation in an on-off fashion. When the a digital signal is used to amplitude modulate a high-frequency carrier, this is called Amplitude Shift Keying (ASK). There are two forms of angular shift keyed modulation; Frequency Shift Keying (FSK) and Phase Shift Keying (PSK). This section shows you how to create all three types of Shift Keyed modulation.

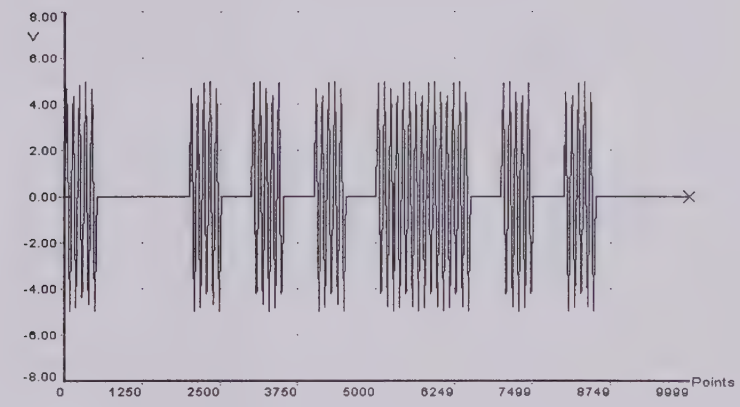

Figure 5: Amplitude Shift Keying

Creating an ASK Waveform

ASK, analogous to Amplitude Modulation (AM) in the world of analog modulation, is the simplest type of keyed digital modulation. A digital signal is used to control the on-off state of a high-frequency carrier. A convenient example of a digital signal is the one we already created in the previous section and stored in file **DIGSIG.CAD**. The WaveCAD equation feature has the unique ability to use .WAV format files as part of an equation. This ability is especially useful for creating modulation signals. The SIN function in the equation feature is used to create the high frequency carrier which is set to 1/100th of the sample rate for this example:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Equation...** from the WaveCAD menu bar.
- Enter the equation:
  Ampl(p) = sin(100*(2*pi*f)*t) * (1024+DIGSIG.WAV)/2558.

- In the **Level Adjuster**, set the **Amplitude** to 10 Volts and click on **Adjust**.
- Click on the **Generate** button.

To view the details of the ASK waveform, WaveCAD's zoomed views may be used. To zoom in on a waveform, use the mouse to directly select the area to be viewed. The WaveCAD graph will then be updated showing the selected area. In addition, a smaller "Waveform Map" window will also appear which shows the selected area in relation to the entire waveform. The "white" section of this window may be dragged to re-locate the WaveCAD viewport. The WaveCAD graph may be zoomed into many times to further magnify the details of a waveform. Select **View/View All** to return to the original view of the waveform.

Creating an FSK Waveform

FSK involves switching a carrier between two different frequencies at the same instant that a digital signal transitions from one level to another. The equation used to create FSK is the equation for the carrier frequency plus and offset frequency that occurs in sync with a digital transition. Once again, the **DIGSIG.WAV** file is utilized to simulate a digital signal in the FSK equation. The **sin** function in the equation feature is used to create the high frequency carrier which shifts between 1/100th and 1/200th of the sample clock for this example:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 10000.
- Click on the **OK** button.

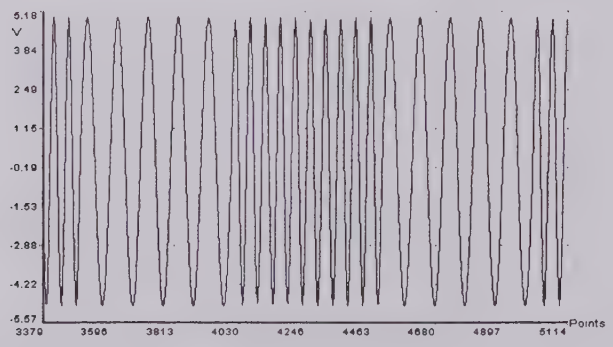

Figure 6: Frequency Shift Keying

- Select **Wave/Equation...** from the WaveCAD menu bar.

- Enter the equation:
  Ampl(p) =sin((100+100*(1535+DIGSIG.WAV)/2558)*(2*pi*f)*t).
- In the **Level Adjuster**, set the **Amplitude** to 10 Volts and click on **Adjust**.
- Click on the **Generate** button.

Creating a PSK Waveform

PSK differs from FSK since the instantaneous phase of the carrier is being shifted instead of the instantaneous frequency. A digital signal is used to control the phase of a high-frequency carrier. Again we will use **DIGSIG.CAD** to simulate a digital signal. When the waveform is a logic "0," the signal has no phase shift and when the waveform is a logic "1," the signal is shifted in phase by 180°. The SIN function in the equation feature is used to create the high frequency carrier which is set to 1/100th of the sample clock and the phase for this example:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Equation...** from the WaveCAD menu bar.
- Enter the equation:
  Ampl(p) =sin(100*(2*pi*f)*t+((1535+DIGSIG.WAV)/2558)*pi).
- In the **Level Adjuster**, set the **Amplitude** to 10 Volts and click on **Adjust**.
- Click on the **Generate** button.

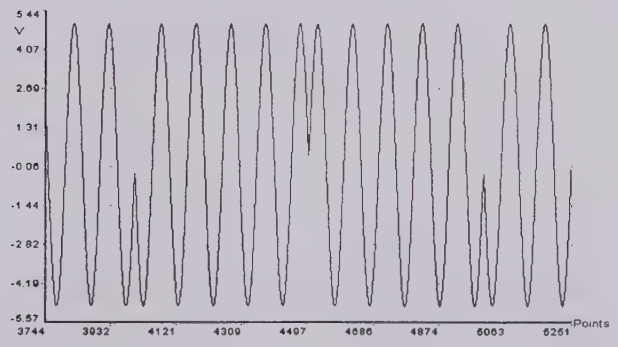

Figure 7: Phase Shift Keying

Loading a Shift-Keyed Waveform From a File

To load a previously saved version of an amplitude shift-keyed waveform like what was created above:

- Select **File/Open...** from the WaveCAD menu bar.

- Select **ASK.CAD** from the list box and click on **OK**.
- An ASK waveform is displayed.

To load a previously saved version of an angular shift-keyed waveform like what was created above:

- Select **File/Open...** from the WaveCAD menu bar.
- Select **FSK.CAD** from the list box and click on **OK**.
- An FSK signal is displayed.
- Select **File/Open...** from the WaveCAD menu bar.
- Select **PSK.CAD** from the list box and click on **OK**.
- A PSK signal is displayed.

Downloading Shift-Keyed Waveforms to a Model 3151 (H/W Demo only)

First, either create a waveform or load a waveform stored in a file as shown above. A waveform should appear graphically. To download the waveform displayed on the WaveCAD graph:

- Select **Setup/Instrument Configuration** from the WaveCAD menu bar.
- Set the **Output/Mode** to **User**.
- Set the **Segment** to 1.
- Check the box next to **Output on/off** to turn on the 3151's output.
- Click on the **Download** button.
- Click on the **Cancel** button.
- Click on the **Waveform/Parameter Download** icon.

A repetitive waveform identical to that displayed in the WaveCAD graph will be displayed on the oscilloscope. Be sure to adjust the frequency and amplitude controls on the oscilloscope to view the waveform properly.

Analog modulation is used in analog communications systems like traditional TV and Radio broadcasts. Amplitude Modulation (AM) and a generalized form of Frequency Modulation (FM) and Phase Modulation (PM) are commonly used for radio broadcasts while Vestigial Sideband Modulation (VSB), which is a type of AM, is commonly used for television broadcasts. Examples of AM and PM are presented below for the case when the high frequency carrier is being modulated by a repetitive waveform (for simplicity) but may be modified to modulate almost any analog signal.

Creating an AM Waveform

AM is the simplest type of analog modulation. In AM, the amplitude of a high-frequency carrier is modulated by a low-frequency signal which contains analog information such as an audible tone. The SIN function in the equation feature is used to create the high frequency carrier which is set to 1/1000th of the sampling period as well as the low frequency modulation tone which is set to 1/2 of the sampling period for this example:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Equation...** from the WaveCAD menu bar.
- Enter the equation:
  Ampl(p) =sin(1000*2*pi*f*t) * sin(2*omg*t).
- In the **Level Adjuster**, set the **Amplitude** to 10 Volts and click on **Adjust**.
- Click on the **Generate** button.

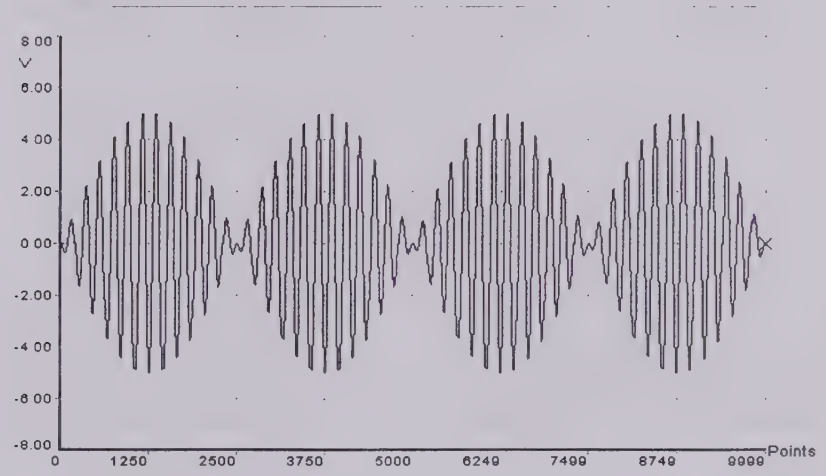

Figure 8: Amplitude Modulated Waveform

Creating PM Waveforms

A repetitive triangle wave will be used to demonstrate PM to make the PM waveform easy to understand. Again, the **sin** function in the equation feature is used to create the high frequency carrier which is set to 1/20th of the sampling period. The triangular waveform is created using a standard WaveCAD function which is saved to a file and then used as part of an equation:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Sawtooth...** from the WaveCAD menu bar.
- Set the number of **Cycles** to 2 and click on the **OK** button.
- Select **File/Save As...** from the WaveCAD menu bar.
- Set the Filename parameter to **TRIANGLE** and click on **OK**.
- Select **Wave/Equation...** from the WaveCAD menu bar.
- Enter the equation:
  Ampl(p) =sin(20*2*pi*f*t+5*triangle.wav/2047).
- In the **Level Adjuster**, set **Amplitude** to 10 and click on **Adjust**.
- Click on the **Generate** button.

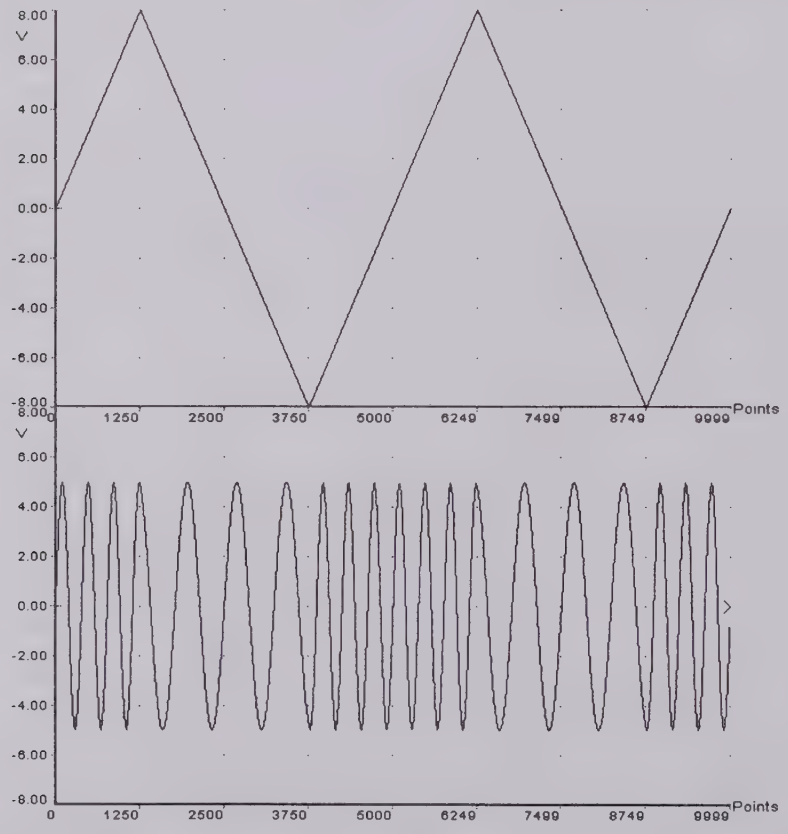

Figure 9: Modulation Source (above) and PM Waveform (below)

Loading a Modulated Waveform From a File

To load a previously saved version of a modulated waveform like those created above:

- Select **File/Open...** from the WaveCAD menu bar.
- Select **AM.CAD** or **PM.CAD** from the list box and click on **OK**.
- A modulated waveform is displayed.

Downloading Modulated Waveforms to a Model 3151 (H/W Demo only)

First, either create a waveform or load a waveform stored in a file as shown above. A waveform should appear graphically. To download the waveform displayed on the WaveCAD graph:

- Click on the **Waveform/Parameter Download** icon.
- Select **Setup/Instrument Configuration** from the WaveCAD menu bar.
- Set the **Output/Mode** to **User**.
- Set the **Segment** to 1.
- Check the box next to **Output on/off** to turn on the 3151's output.
- Click on the **Download** button.
- Click on the **Cancel** button.

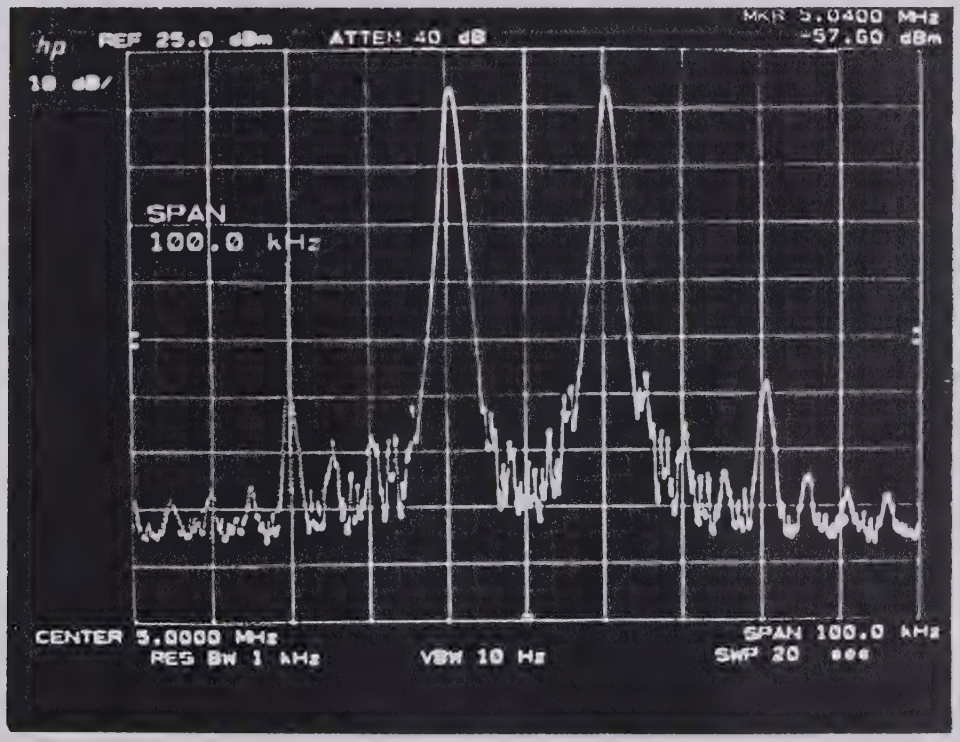

Figure 10: AM Waveform Viewed on a Spectrum Analyzer

A repetitive waveform identical to that displayed in the WaveCAD

graph will be displayed on the oscilloscope. Be sure to adjust the frequency and amplitude controls on the oscilloscope to view the waveform properly. To gain a better understanding of a modulated waveform, view the modulated waveform on a spectrum analyzer. Notice the AM "sidebands" at frequencies above and below the carrier frequency shown in Figure 10.

Sweep signals are used for testing filters, radio equipment, frequency response and for many other general purpose applications. Two types of sweep are commonly used: linear and logarithmic. A linear sweep ramps from its start frequency to its destination frequency with equal weight to all intermediate frequencies. A logarithmic sweep dwells more on the lower frequencies as the frequency changes. Examples of both types of sweep are included below.

Creating a Linear Sweep Waveform

A linear sweep ramps from an initial frequency to a destination frequency which is typically one decade or more away from the initial. For this example, two cycles of a half decade sweep is generated which is increasing in frequency. The WaveCAD **Sawtooth...** function is used to create a ramp function. This function is then incorporated into a WaveCAD equation to control the increase in frequency of a SIN function:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Sawtooth...** from the WaveCAD menu bar.
- Set the number of **Cycles** to 0.25 (producing a ramp only).
- Click on the **OK** button.
- Select **File/Save As...** from the WaveCAD menu bar.
- Set the **Filename** parameter to **RAMP**.
- Click on the **OK** button.
- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Equation...** from the WaveCAD menu bar.
- Enter the equation:
  Ampl(p) =sin((1+ramp.wav/506)*2*pi*omg*t).
- Set the **Cycles** parameter to 2.
- In the **Level Adjuster**, set the **Amplitude** to 10 Volts and click on **Adjust**.
- Click on the **Generate** button.

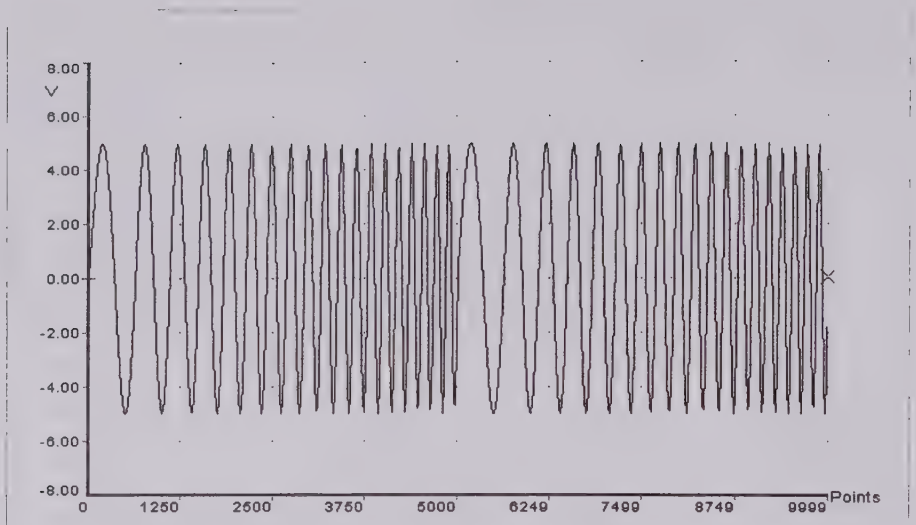

Figure 11: 2 Cycles of a Linear Sweep of a Half Decade

Creating a Log Sweep Waveform

A log sweep ramps logarithmically from an initial frequency to a destination frequency which is typically one decade or more away from the initial. For this example, a multiple decade sweep down in frequency is generated. Note that the frequency at p=10000 is 10% of the frequency p=1000 which is 10% of the frequency at p=100 and so on. This type of sweep enables you to include many frequencies within a reasonably small waveform vector. A WaveCAD equation is used to create the log sweep as follows:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Equation...** from the WaveCAD menu bar.
- Enter the equation:
  Ampl(p) =sin(21.995*log(t+.001)).
- Set the **Cycles** parameter to 2.
- In the **Level Adjuster**, set the **Amplitude** to 10 Volts and click on **Adjust**.
- Click on the **Generate** button.

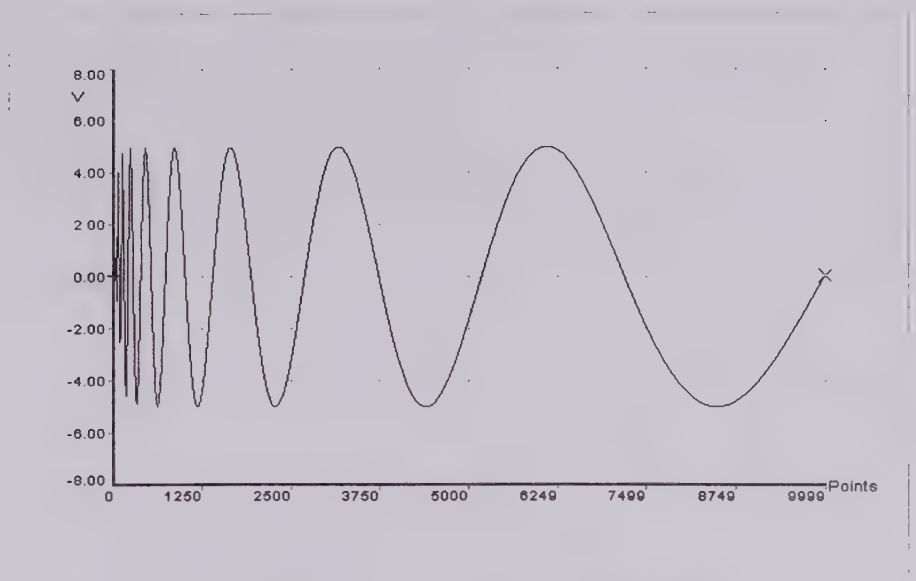

Figure 12: A Multi-Decade Log Sweep of Decreasing Frequency

Loading a Swept Waveform From a File

To load a previously saved version of a linear sweep waveform like what was created above:

- Select **File/Open...** from the WaveCAD menu bar.
- Select **LINSWEEP.CAD** from the list box and click on **OK**.
- Two cycles of a linear sweep waveform is displayed.

To load a previously saved version of a log sweep waveform like what was created above:

- Select **File/Open...** from the WaveCAD menu bar.
- Select **LOGSWEEP.CAD** from the list box and click on **OK**.
- A log sweep signal is displayed.

Downloading Sweep Waveforms to a Model 3151 (H/W Demo only)

First, either create a waveform or load a waveform stored in a file as shown above. A waveform should appear graphically. To download the waveform displayed on the WaveCAD graph:

- Click on the **Waveform/Parameter Download** icon.
- Select **Setup/Instrument Configuration** from the WaveCAD menu bar.
- Set the **Output/Mode** to **User**.
- Set the **Segment** to 1.
- Check the box next to **Output on/off** to turn on the 3151's output.
- Click on the **Download** button.

- Click on the **Cancel** button.

A repetitive waveform identical to that displayed in the WaveCAD graph will be displayed on the oscilloscope. Be sure to adjust the frequency and amplitude controls on the oscilloscope to view the waveform properly.

This section describes how to use WaveCAD's standard wave menu to create multi-phase waveforms. These are useful in cases when the customer intends to use multiple 3151's to control separate phases independently.

Displaying a Multi-Phase Waveform

For this example, a 3-phase sinusoidal waveform is to be modelled as follows:

- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Amplitude** to 16 and **Pts** to 1024.
- Click on the **OK** button.
- Select **Wave/Sine** from the WaveCAD menu bar.
- Click on the **OK** button.
- Select **File/New** from the WaveCAD menu bar.
- Select **Wave/Sine** from the WaveCAD menu bar.
- Change the **Start Phase** parameter to 120.
- Click on the **OK** button.
- Select **Wave/Sine** from the WaveCAD menu bar.
- Change the **Start Phase** parameter to 240.
- Click on the **OK** button.
- Select **Window/Tile** from the WaveCAD menu bar.

Three phases are displayed, each of which is 120° apart.

Modifying a Multi-Phase Waveform

Suppose that this 3-phase waveform is controlling the 3 phases of an AC power source and the user would like to simulate a power sag in phase #2. WaveCAD's freehand drawing feature can be used to do this as follows:

- Click in the window with the 0° phase shift and maximize this window.
- Click on the **Draw Freehand Waveform** icon.
- Hold down the mouse button to draw a sag at the peak of the waveform.
- Click on the **Draw Freehand Waveform** icon.
- Select **Window/Tile** from the WaveCAD menu bar.

Loading Multi-Phase Waveforms From a File

To load multi-phase waveforms:

- Select **File/Open...** from the WaveCAD menu bar.
- Select **PH0.CAD** from the list box and click on OK.
- Select **File/Open...** from the WaveCAD menu bar.

- Select **PH120.CAD** from the list box and click on OK.
- Select **File/Open...** from the WaveCAD menu bar.
- Select **PH240.CAD** from the list box and click on OK.
- Select **Window/Tile** from the WaveCAD menu bar.

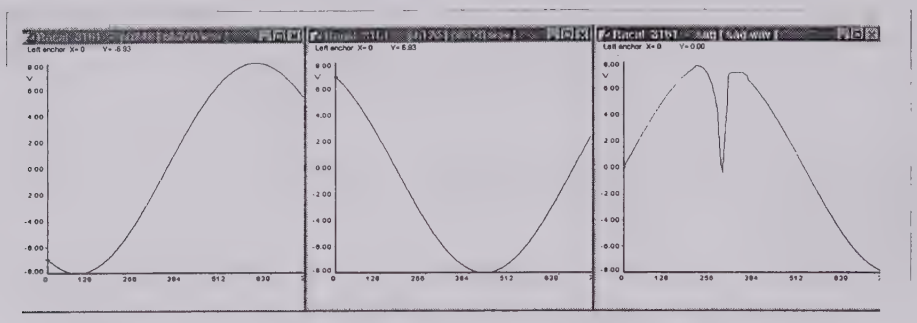

Figure 13: Tiled 3-Phase Waveform with One Sagging Phase

Downloading to Multiple 3151's (H/W Demo Only)

The hardware setup described in the "Getting Started" section will have to be modified to include three 3151's at three different logical addresses. For simplicity, try to use addresses 2-4. If those are not available then you may choose others but this example will presume the use of addresses 2-4. In addition, an oscilloscope with at least three channels will be required to view all phases simultaneously. First, configure the master 3151 at address 2 (or other):

- Click in the **PH0.CAD** window and maximize this window.
- Select **VXI/Address & Options...** from the WaveCAD menu bar.
- Set the address to 2 (or the actual address of the first 3151).
- Select **VXI/Command Editor...** from the WaveCAD menu bar.
- In the **Edit Field,** enter **PHASe:LOCK ON** and click on **Add**.
- In the **Edit Field**, enter **PHASe:SOURce MASTer** and click on **Add**.
- The Message List should now list these two commands.
- Click on the **Download** button, then close the Command Editor window.
- Select **VXI/Download Wave/Parameters** from the WaveCAD menu bar.
- Select **Setup/Instrument Configuration** from the WaveCAD menu bar.
- Check the box next to **Output On** to turn on the 3151's output.
- Click on the **Download** button.
- Click on the **Cancel** button.

Next, configure the 3151's at addresses 3 and 4 (or other) as

slaves:

- Select **VXI/Address & Options...** from the WaveCAD menu bar.
- Set the address to 3 (or the actual address of the second 3151).
- Select **VXI/Command Editor...** from the WaveCAD menu bar.
- In the **Edit Field,** enter **PHASe:SOURce SLAVe** and click on **Add**.
- In the **Edit Field,** enter **PHASe:LOCK ON** and click on **Add**.
- The Message List should now list these two commands.
- Click on the **Download** button, then close the Command Editor window.
- Select **VXI/Download Wave/Parameters** from the WaveCAD menu bar.
- Select **Setup/Instrument Configuration** from the WaveCAD menu bar.
- Check the box next to **Output on/off** to turn on the 3151's output.
- Click on the **Download** button.
- Click on the **Cancel** button.
- Select **VXI/Address & Options...** from the WaveCAD menu bar.
- Set the address to 4 (or the actual address of the third 3151).
- Select **VXI/Command Editor...** from the WaveCAD menu bar.
- In the **Edit Field**, enter **PHASe:SOURce SLAVe** and click on **Add**.
- In the **Edit Field**, enter **PHASe:LOCK ON** and click on **Add**.
- The Message List should now list these two commands.
- Click on the **Download** button, then close the Command Editor window.
- Select **VXI/Download Wave/Parameters** from the WaveCAD menu bar.
- Select **Setup/Instrument Configuration** from the WaveCAD menu bar.
- Check the box next to **Output on/off** to turn on the 3151's output.
- Click on the **Download** button.
- Click on the **Cancel** button.

Repetitive waveforms identical to those displayed in the WaveCAD graphs will be displayed on the oscilloscope. Be sure to adjust the frequency and amplitude controls of the oscilloscope to view the waveforms properly.

One particular navigation application required the creation of "huge" waveforms which require waveforms which last for seconds but contain features requiring the entire frequency resolution of the 3151. This "huge" waveform will therefore require different techniques than those discussed thus far to avoid running out of waveform memory (a 5 second waveform would require 500 Megasamples of memory with the sample clock at 100MHz).

Creating a Sequence

This application requires the creation of a pulse waveform that is "high" (1.5V) for 3 seconds and "low" (0V) for 2 seconds and then repeats. The application requires the sample clock to run at 100MHz to create allow the creation of short delays with respect to other instruments running in sync. Normally, the creation of a 5 second waveform clocked at 100MS/s would require 500 Megasamples; the 3151 has up to 512k of sample memory. To overcome this shortfall, waveform sequences are used with waveform looping. Thus, short waveforms stored in memory segments are then repeated thousands of times to create huge waveforms with modest amounts of memory.

This process involves defining the waveform segments required for the sequence and then defining the sequences. Although segments and sequence lists are displayable in WaveCAD, the resulting sequences are not. The only way to view the sequence created in this section is with a scope. The procedure is as follows:

- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change **Pts** to 10000.
- Click on the **OK** button.
- Select **Wave/Square...** from the WaveCAD menu bar.
- Set **Cycles** to 1.5 and **Amplitude** to 3.
- Click on the **OK** button.
- Slide the left anchor to 1238 and the right anchor to 8742.
- Click on the **Draw Straight Line Segments** icon.
- Click on the **Remove Anchors** icon.
- Select **File/Save As...** from the WaveCAD menu bar.
- Set the **Filename** parameter to **1P5.CAD**.
- Click on the **OK** button.
- Select **File/New** from the WaveCAD menu bar.
- Select **Setup/Axis...** from the WaveCAD menu bar.
- Change the **Sampling Clock** to 100e+6 Hz.
- Click on the **OK** button.

- Select **File/Save As...** from the WaveCAD menu bar.
- Set the **Filename** parameter to **0P0.CAD**.
- Click on the **OK** button.
- Select **Setup/Sequence Table/New Sequence...** from the WaveCAD menu bar.
- Edit Link 1 setting **Loop** to 300000 and **Waveform file** to **1P5.WAV**.
- Click on the **Add/Replace** button.
- Edit Link 2 setting **Link** to 2, **Loop** to 1 and **Waveform file** to **0P0.WAV**.
- Click on the **Save** button and then close the dialog box.

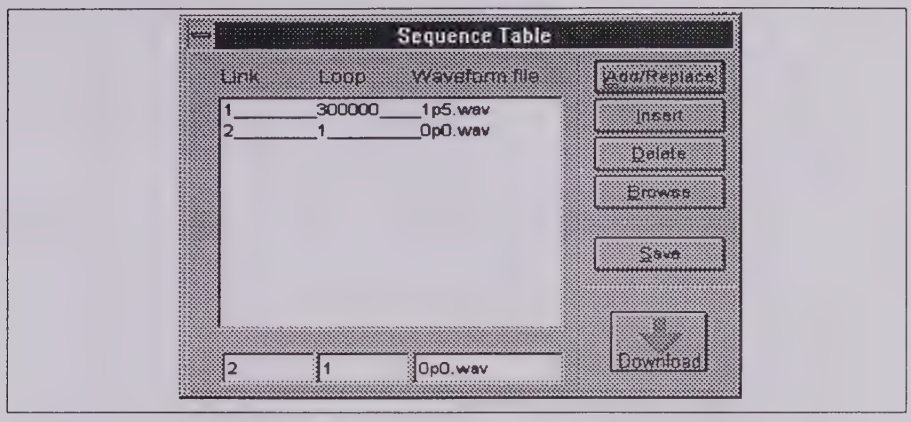

Figure 15: Sequence Table for this Example

A sequence has been defined that stays at 1.5 Volts for 300,000 cycles of a 1000 point segment which lasts 1000 x 10ns = 10µs. The loop parameter multiplies this period by 300,000 to get a total high time of 300,000 x 10µs = 3s. Immediately after this, the output will go to 0V and remain there even after the 10µs period (as long as the 3151 is not in continuous mode). To set the repetition interval to 5 seconds, the internal trigger timer is used.

Downloading a Sequence

The steps below explain how to set the internal trigger timer to a 5 second period and download the sequence to the 3151:

- Click on the **Download Current Sequence** icon.
- Select **Setup/Instrument Configuration** from the WaveCAD menu bar.
- Check the box next to **Output on/off** to turn on the 3151's output.
- Set the Output Mode to **Sequence**.
- Set the **Int. Per.** to 5 seconds.
- Set **Trigger Mode** to **Trig**.
- Click on the **Download** button.

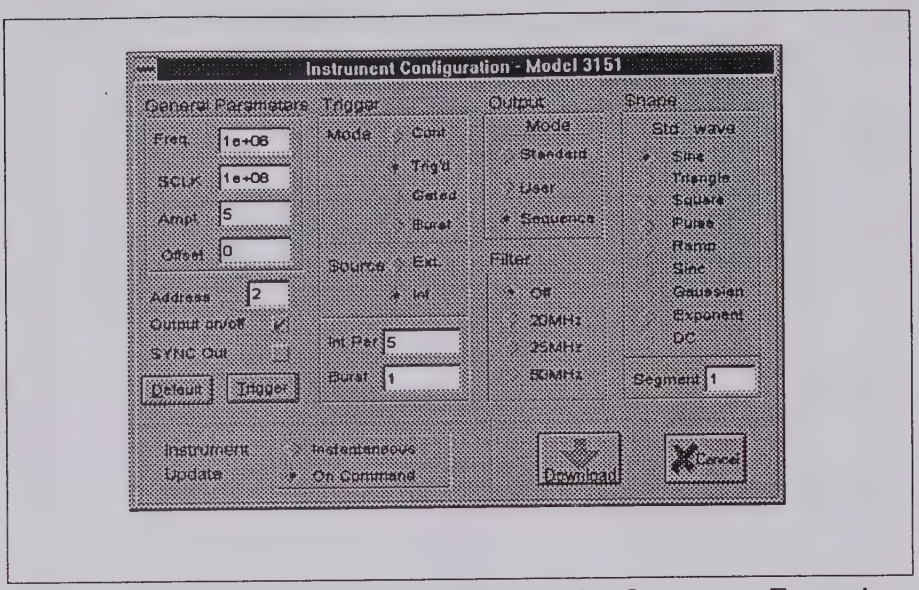

Figure 16: Instrument Configuration for the Sequence Example

Amplitude Modulation (AM) - A high frequency carrier modulated in amplitude by a signal containing information.
Amplitude Shift Keying (ASK) - A high frequency carrier modulated in amplitude by a digital signal containing information.
Analog Signal - A signal with continuous voltage levels.

Digital Signal - A signal with discrete voltage levels.

Excel - A spreadsheet program from Microsoft.

Frequency Modulation (FM) - A high frequency carrier whose frequency is modulated by a low frequency signal containing analog information.
Frequency Shift Keying (FSK) - A high frequency carrier whose frequency is modulated by a low frequency signal containing digital information.

Huge Waveform - A very long waveform with ultra-fine details.

Linear Sweep - A sweep in the frequency of a waveform in which the frequency increases (or decreases) according to a ramp function.
Logarithmic Sweep - A sweep in the frequency of a waveform in which the frequency increases (or decreases) according to a logarithm.
Loop - The repetition of a waveform segment from 1 to 1Meg times.

Mathcad - A "word processor" for math from Mathsoft.
Microsoft Windows - Menu based graphical operating system; version 3.1 and Windows 95 are compatible with WaveCAD.

Noise - Random energy which may be added to any signal.

Phase Modulation (PM) - The phase of a high frequency carrier is modulated by a signal containing information.
Phase Shift Keying (PSK) - The phase of a high frequency carrier is modulated by a digital signal containing information.

Segment - A 3151 memory trace containing a waveform of at least 10 points and containing an even number of points.
Sequence - A series of waveform segments which are output contiguously by a 3151.
Sweep - A signal which increases or decreases in frequency over time.

Three-Phase Waveform - Three sine waves typically 120° apart.

WaveCAD - Arbitrary waveform creation and 3151 control software available from Racal Instruments.

Vestigial Sideband Modulation (VSB) - A form of AM which combines the properties of both Single Sideband (SSB) and Double Sideband (DSB) modulations. Instead of cutting off one sideband completely, like SSB, a filter is used to roll the sideband off gradually such that the signal spectrum near the carrier is transmitted partially in both sidebands and the signal spectrum farther away from the carrier is transmitted on only one sideband. Commonly used for commercial television broadcasting.

VXIbus - VMEbus Extension for Instrumentation Revsion 1.4 as defined by the VXIbus consortium.

Amplitude Modulation (AM) - 15,17,18,29
Amplitude Shift Keying (ASK) - 11,12,14,29
Analog Signal - 9,10,29

Digital Signal - 8,9,29

Frequency Modulation (FM) - 15,29
Frequency Shift Keying (FSK) - 11-14,29

Huge Waveform - 26,29

Icons - 4

Linear Sweep - 19,21,29
Logarithmic Sweep - 20,21,29

Noise - 7-10,29

Phase Modulation (PM) - 15-17,29
Phase Shift Keying (PSK) - 11,13,14,29

Sequence - 26-29
Sweep - 19-21,29

Vestigial Sideband Modulation (VSB) - 15,30
VXIbus - 1,2,8,30

Waveform Map - 12

Zoom - 12

MP MASTER
MA31116